Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 121 (2015) 1726 – 1733

Procedia Engineering

www.elsevier.com/locate/procedia

9th International Symposium on Heating, Ventilation and Air Conditioning (ISHVAC) and the 3rd International Conference on Building Energy and Environment (COBEE)

Application of Field Synergy Principle for Fin Reshaping of a Natural Convection Radiator

Yu Zhang[a,]*, Xiaohua Liu[a]

[a]*Building Energy Research Center, Tsinghua University, Beijing, 100084, China*

Abstract

An optimized shape of the fin could enhance the heat transfer for a certain convection radiator while saving material. Based on the validated CFD simulation, this research change the two dimensional shape of fin based on field synergy principle analysis with the synergy angle as the monitor for the level of synergy. The analysis shows that a shape group pair of “cross-V” is better than the traditional square fin. The convective heat transfer coefficient is improved to 14.87 W/ (m^2K) comparing to 2.75 W/ (m^2K) of the original model with the square fin. Also, the back part of the fin could be removed off for a better air flow consideration. The heat transfer coefficient increase with the average synergy angle.

Peer-review under responsibility of the organizing committee of ISHVAC-COBEE 2015

Keywords: Field synergy principle, Computational fluid dynamics, Shape of the fin, Heat transfer enhancement, Natural convection radiator

1. Introduction

Convective heat transfer enhancement is always a fundamental and everlasting question in terms of higher efficiency and metal saving for a heat exchanger. A lot of work have been done on this subject from academic to industry. A most widely acceptable classification of heat transfer enhancement techniques is the method of Bergles et al.’s [1, 2]. In the literature review, the most techniques can be divided into passive techniques and active techniques [2]. During the recent years, many new techniques have been applied into reality, like the moving walls and jet [3]. However, it remains effective for heat transfer enhancement to enlarge the heat exchange area and

* Corresponding author. Tel.: +86-13810532049
E-mail address: thuzyu19@gmail.com

Peer-review under responsibility of the organizing committee of ISHVAC-COBEE 2015
doi:10.1016/j.proeng.2015.09.142

destroy the thermal boundary layer. As for the passive techniques, the high intensity heat exchanger and complex three-dimensional fin geometry or treatments are commonly the directions. Hence a simple way to reduce the heat exchange area, improving the heat transfer at the same time is of high practical values. The optimization of the two-dimensional shape of its fin benefits the industry for this saves material and is easier to be modified.

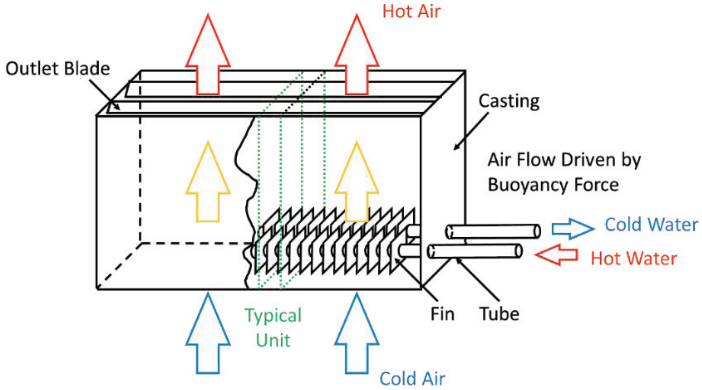
Guo's Field Synergy Principle (FSP) [4] give the guidance on the direction of heat transfer enhancement. The intersection angle between the velocity and the temperature gradient field should be closer to 0 degree to get a higher heat transfer coefficient. The key steps for proposal can be concluded in other researches [4, 5]. Computational fluid dynamics is adopted when it comes to the application of FSP. Yu. Jin et al. [6] set seven geometry parameters. The research reveals that the Nu number is in good cooperation with the parameters and FSP based on the simulation. M.O.A. Hamid et al. [7] work on the heat transfer enhancement of the pre-heater in a solar-assisted MED desalination unit. It comes out as a conclusion that the heat transfer is enhanced significantly by increasing the field synergy number and decreasing of the synergy angle. L.T. Tian et al. [8] perform the simulation on the laminar heat transfer and fluid flow character of a flat-plate channel with longitudinal vortex generators. The longitudinal vortex generators increase the heat transfer by reduction of the intersection angle between velocity and temperature gradient.

Natural convection radiator is commonly utilized as the heating terminal in winter. This paper will focus on the optimization of the fin shape, and FSP will be adopted as the guidance for the fin reshaping.

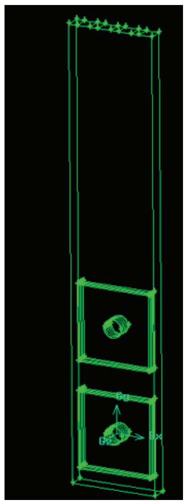
2. Simulation and validation

2.1. Model of the natural convection radiator

The structure of a typical natural convective radiator is shown in Fig 1(a). The nature convection heat transfer is dominating in such an air-water heat exchanger. The material of the casting and fin is steel and aluminum separately. The material of the tube is copper, while the surface of the tube was covered by a thin aluminum casting from the fin. The heat transfer process can be simulated by FLUNET software, and the preprocessor GAMBIT can be used to generate the mesh needed.



(a)



(b)

Fig. 1. A typical natural convective radiator. (a) The structure; (b) The structure of the original model in Gambit.

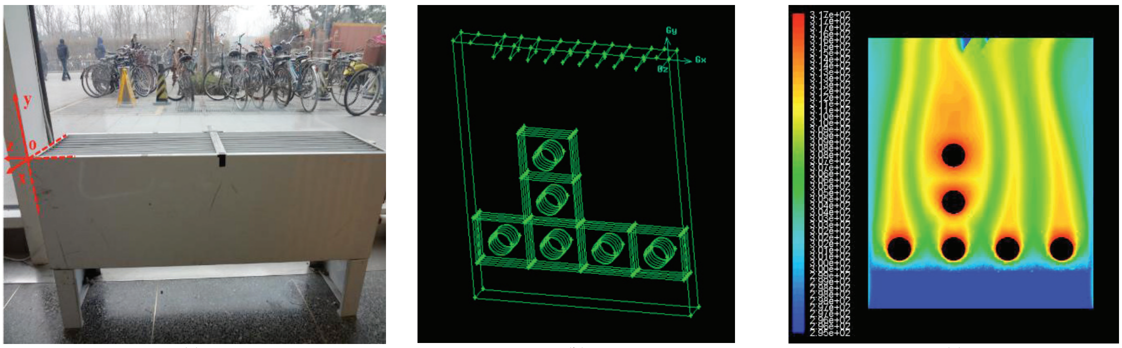
A hypothesis, "typical unit" as shown in Fig 1(b), is made that the situation of heat transfer and fluid field of the typical unit could represent the heat transfer process of other space of the heat exchanger. The front and back surfaces of the computation zone is the casting wall. The left and right surfaces should be the symmetry wall. The down and up surface is for air flow. Some outlet blades are placed near the up surface to control the air flow direction as the same in reality. As for the casting wall, the wall is assumed to be thermally insulated, which refers to that the convective heat transfer and radiation of the casting wall could be neglected compared to the heat transfer of the fin. To simplify the modeling, the temperature of the tube is simplified to be fixed. This may be a little controversial. The temperature of the inlet and out is different obviously. We can enlarge the flow rate of the water in the pipe to decrease the temperature difference between the inlet and outlet of the tube. So the constant temperature for the tube is the ideal condition, which will contribute to the symmetry.

In the simulation, the density of the air is computed as ideal gas, which is acceptable for natural convection. The RNG $k-\varepsilon$ model is adopted with the energy equation on. The SIMPLEC algorithm is used under the condition of the coupling of pressure and velocity. The second-order upwind scheme is employed. The conditions for boundary are stated as follows [9]. The left and right surfaces are symmetry wall to better imitate the repeatable unit in the middle part of the radiator. The front and behind wall is set as thermal insulated. The up wall is pressure outlet of 0 Pa. The down wall is pressure inlet of 0 Pa. To better simulate the temperature distribution of the fin, the User Defined Function (UDF) is interpreted. To calculate the synergy angle between velocity and temperature gradient [4], a Custom Defined Function is adopted according to equation (1).

$$\mathrm{Re}_x \,\mathrm{Pr}\int_0^1 (\vec{\mathrm{U}} \bullet \nabla\overline{T})d\,\overline{y} = Nu_x \tag{1}$$

2.2. Validation of the CFD results

The heat transfer performances of a typical heat exchanger in Tsinghua University, shown in Fig 2(a), were measured and used for the validation of the CFD simulation results. The width, height and length of the casting is 0.25m, 0.5m and 1.0 m respectively. The thickness of the fin is 0.0003m, and the fin spacing is 0.005m. This radiator has six tubes to form a “T” upside-down partly as shown in Fig 2(b). A space Cartesian system is adopted. During the on-site measurement, 42 temperature points are placed on z=-0.625m plane to measure the inside air temperature. The point for temperature of the room is set at the center of the room with the height of 1.50 m. The average temperature for the water tube inlet and outlet is 318.6K and 314.9K respectively.



(a) (b) (c)

Fig. 2. The validation experiment and simulation. (a) The convective radiator with space Cartesian coordinate system; (b) The outline structure of the real heat exchanger; (3) The contour of temperature distribution of the mid plane between two fins.

The simulation assumption and procedure are the same as described in the above section. The casting is kept in the simulation, due to that the natural convection will be enlarged. The simulation results of the temperature distribution within the radiator are shown in Fig 2(c). And the experiment data and the simulation result are compared in Fig 3.

The highest temperature measured is 311.9K while the lowest is 305.5K. The average absolute deviation of the simulated and measured values is 1.3K. The points with relatively large deviation, as compared with Fig 2(a), are all near the front and the back wall and near the corner. This is reasonable. There is a rod inside the casting near the corner, which is neglected in the simulation. And the complex curly outlet blades are simplified as inclined flat surface shown in Fig 2(b).

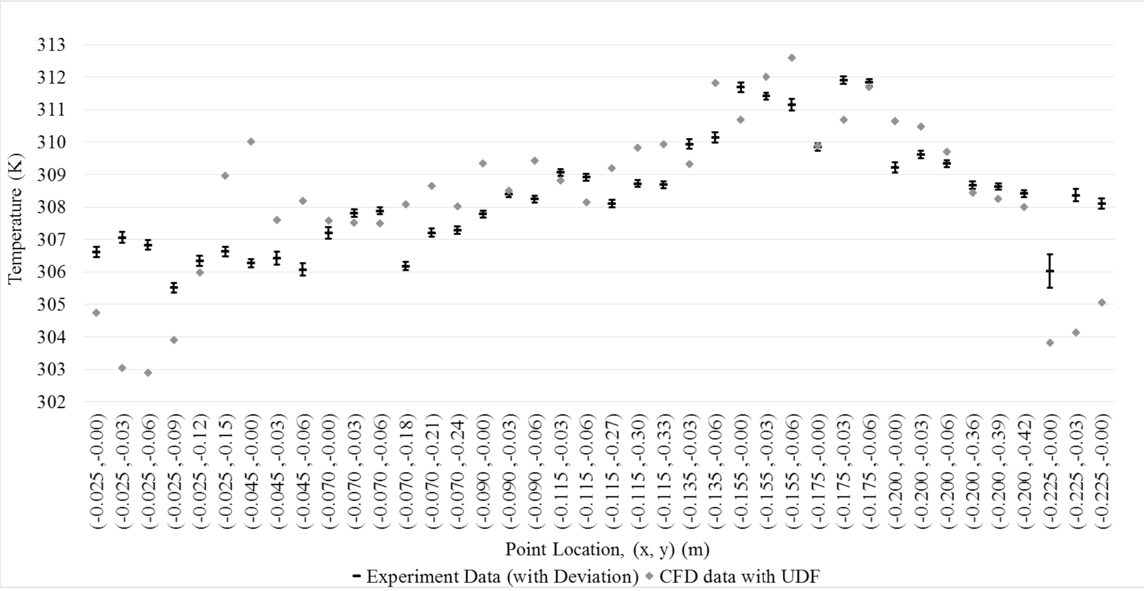
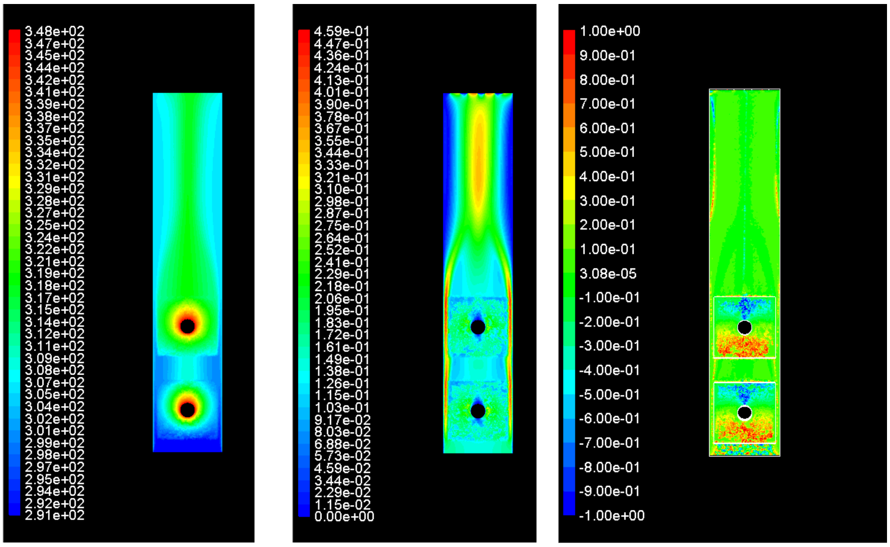


Fig. 3. Difference between measured temperature and CFD model value (z=-0.0625m).

3. Original radiator performance

The validated CFD method is then utilized to investigate the performance of the natural convection radiator shown in Fig 4. The model is simplified to two tubes shown as indicated in Fig 4(b). One tube is in the top while the other down. We calculate 5 fins rather than 2 on a tube to form a typical unit. The main geometry is built according to reality shown in Table 1. The casting of the convective radiator will enhance the heat transfer, the height of the casting in the model is set as 0.6 m and the temperature for the tube is raised to 348K. This model is named as original model. The number of the grid is approximately 1.5 million. Four boundary layers are placed near the fin surface.



(a) (b) (c)

Fig. 4. The mid plane between two fins of the original model. (a) Contour of temperature distribution; (b) Contour of velocity; (c) Contour of cosine of the intersection angle between the velocity and the temperature gradient.

Table 1. The geometry parameters of the original model.

Geometry parameter	Casting width (x axis)	Casting height (y axis)	Casting length (z axis)	Fin length	Fin width	Fin thickness	Fin spacing	Diameter of the tube
Model value (m)	0.12	0.6	0.02	0.1	0.1	0.0005	0.004	0.026

The total heat transfer and mass average convective heat transfer coefficient are the most important parameters when evaluating the ability of such a nature convection radiator. The total heat transfer is calculated based on heat capacity and mass flow rate as shown in Table 2. The convective heat transfer coefficient of the original model is 2.75 W/ (m^2K). Usually, this parameter for nature convection is from 3W/ (m^2K) to 10 W/ (m^2K) [2]. The average intersection angle of the mid plane between two fins is 88.2°. The optimization direction is to reduce the intersection angle through changing the fin shape.

Table 2. Numerical results the three models.

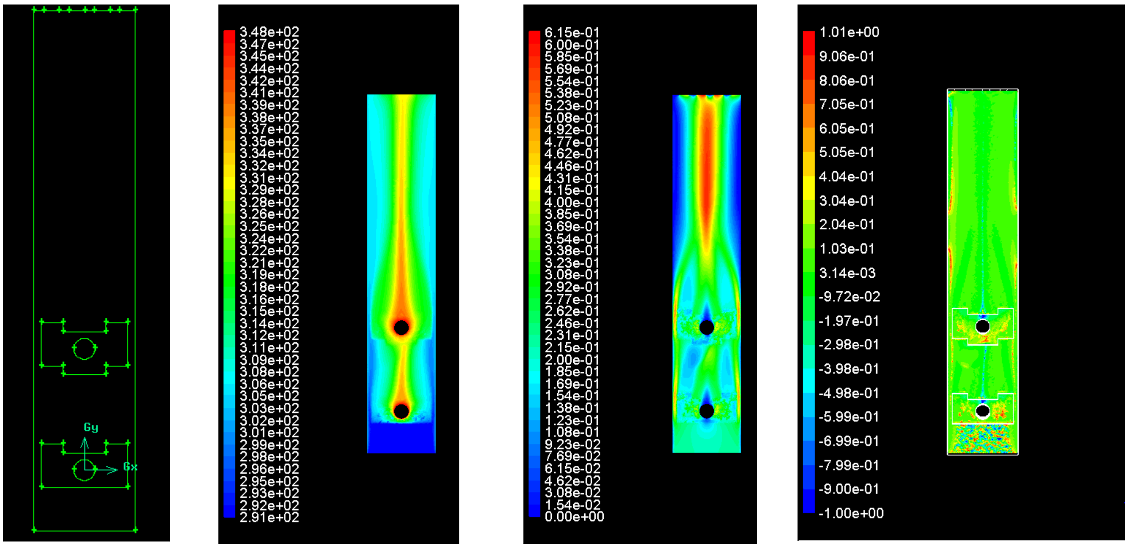
Model	Original Model (Fig.4)	Improved Model (Fig.5)	Final Model (Fig.6)
The area of the fin (double sided) on the up tube (m^2)	0.0189	0.0089	0.0040
The area of the fin (double sided) on the down tube (m^2)	0.0189	0.0079	0.0057
Mass average cosine of the intersection angle of the mid plane between two fins	0.032	0.036	0.050
Average of the y velocity of the air outlet (m/s)	0.300	0.416	0.522
Average of the temperature of the air outlet (K)	312.4	320.2	319.0
Average of the density of the outlet (kg/m^3)	1.130	1.103	1.107
Total heat transfer of the model (W)	16.97	31.27	37.82
Average convective heat transfer coefficient ($W/(m^2K)$)	2.75	7.58	14.87

4. Improvement and analysis

The aim of the optimization is to improve the cosine value. The cosine is positive usually when the heat transfer from the fin and tube to the fluid like the front part of the fin in Fig 4(c). As for the negative situation, there are two possibilities. One is that the heat spread and debilitate to the further region of the fluid by vortex. The other one will deteriorate the heat transfer, that heat transfer from the fluid to fin or casting like the back part of the fin. The cosine of the middle part of the fin is near 0, which will not contribute to heat transfer nor deteriorate it.

4.1. Improvement and improved model

The temperature distribution of the air near the down tube is similar to that of the fin to a great extent as shown in Fig 4(b). It is reasonable to cut off the back part of the fin to avoid the blue part in Fig 4(d). There is a large part of the fin where the cosine value is around 0. So it is acceptable to reduce the area of the front part of the fin to indirectly reduce the green space. The corner of the front part of the fin is of low cosine value. It is acceptable to remove the corner. According to the analysis above, the improved model can be obtained shown in Fig 5(a). The heat transfer performance of the improved model is summarized in Table 2. The area of the fin (double sided) is reduced from 0.0189m^2 to 0.0089m^2 for the up tube and from 0.0189m^2 to 0.079m^2 for the down tube. The average cosine increase a bit, from 0.032 of the original model to 0.036. The convective heat transfer coefficient is improved, from the 2.75 W/ (m^{2}K) of the original model to 7.58 W/ (m^{2}K).



(a) (b) (c) (d)

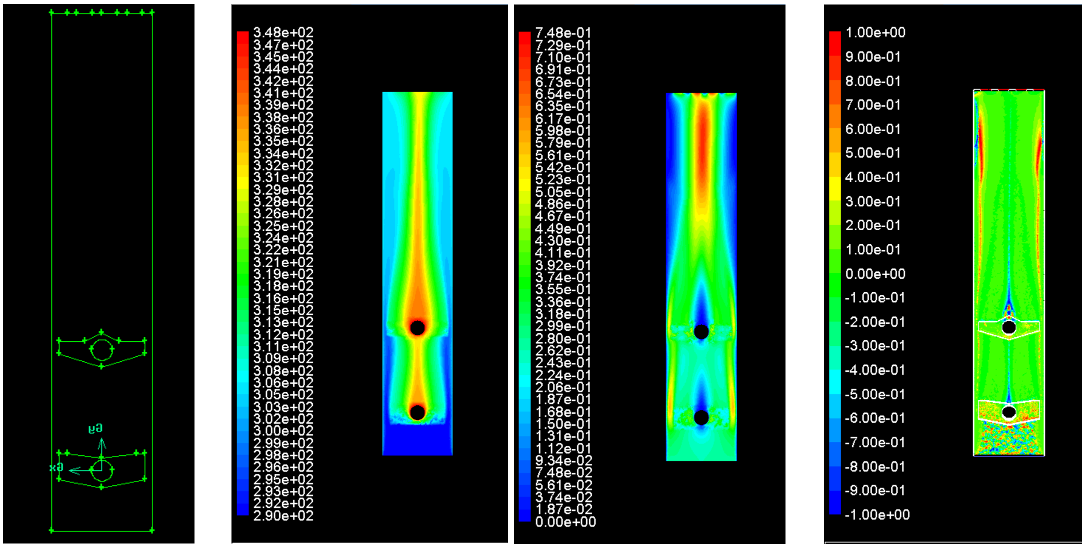
Fig. 5. The mid plane between two fins of the improved model. (a) Overall structure; (b) Contour of temperature distribution; (c) Contour of velocity; (d) Contour of cosine of the intersection angle between the velocity and the temperature gradient.

4.2. Improvement and final model

According to the simulation results of Fig 5, this modification doesn't work well with the fin on the up tube for the improved model. Most of the back part of the fin is still in green and blue. This may be explained by Fig 5(c) and 5(d). The air is slowed by the rest of the back part of the fin, which lead to the vast space with a low cosine. So the back part of the fin on the top tube is totally removed. Also, small modification for the front corner is applied to have a better shape. Finally, we get the final model shown in Fig 6(a). The heat transfer performance of the final model is summarized in Table 2. The area of the fin is reduced from 0.0089m^2 to 0.0040m^2 (double sided) for the up tube and from 0.0079m^2 to 0.057m^2 for the down tube, which is beneficial for industrial because of material saving.

The heat transfer is obviously enhanced. There is a connection between the intersection angle and the heat transfer coefficient. The mass average of cosine of the intersection angle of the mid plane between two fins is improved from 0.032 of original model to 0.036 of improved model and finally to 0.050 of final model. The mass average convective heat transfer coefficient is improved as mentioned above. Both two are increasing after the optimization and increase a lot compared to the original one. It is acceptable to conclude that the convective heat transfer coefficient will increase with the cosine of the intersectional angle if other conditions are similar. By application of field synergy principle, a new two-dimensional shape group of fins, the "cross-V" shape alike pair in improved model, is found and demonstrated.

The convective heat transfer coefficient of the original model is 2.75 W/ (m^2K). After the improvement, this parameter is improved to 7.58 W/ (m^2K) and finally to 14.87 W/ (m^2K). Also, the total heat transfer of the model is largely increased from 16.97W to 37.82W.



(a) (b) (c) (d)

Fig. 6. The mid plane between two fins of the final model. (a) Overall structure; (b) Contour of temperature distribution; (c) Contour of velocity; (d) Contour of cosine of the intersection angle between the velocity and the temperature gradient.

5. Conclusion

By application of the field synergy principle, the shape of a natural convection radiator with copper-tube and aluminum-fin is well analyzed in this paper. A measurement experiment is done to validate the CFD model. By optimizing in FLUENT software, some better design of fins is put out. The main conclusions are as following.

A new two-dimensional shape of the fin, the "cross-V" for a convective radiator is put out. The convective heat transfer coefficient is improved to 14.87 W/ (m^2K). Comparing to 2.75 W/ (m^2K) of the original model with the square fin, the new shape of fin has improve the heat transfer to a great extent and the area of the fin is reduced to 25.6% of the original design.

The cosine of the intersectional angle is used in this research as the monitor to reflect the level of field synergy. There is a connection between convective heat transfer coefficient and the cosine of the intersection angle. With other conditions remaining similar and stable, the convective heat transfer coefficient will increase with the cosine of the intersection angle. The field synergy principle is very helpful with guidance under the condition of this research.

It is reasonable and well demonstrated that the back part of the fin could be removed off for a better air flow situation, which is helpful for heat transfer enhancement. Also, the corner face to the air flow could be removed. This conclusion works under the condition of convective radiator with limited numbers of tubes.

References

[1] R.L. Webb, A.E. Bergles, Heat transfer enhancement: second generation technology, Mechanical Engineering, 115(6) (1983) 60-67.
[2] A.E. Bergles, Some perspectives on enhanced heat transfer second Generation heat transfer technology, ASME Journal of Heat Transfer, 110(1988) 1082-1096.
[3]L. Léal, M. Miscevic, P. Lavieille, M. Amokrane, F. Pigache, F. Topin, B. Nogarède, L. Tadrist, An overview of heat transfer enhancement methods and new perspectives Focus on active methods using electro active materials, Int. J. Heat Mass Tran. 61 (2013) 505–524.
[4]Z.Y. Guo, D.Y. Li, B.X. Wang, A novel concept for convective heat transfer enhancement, Int. J. Heat Mass Transfer[J] 41 (1998) 2221–2225.
[5]Wen-Quan Tao, Zeng-Yuan Guo, Bu-Xuan Wang. Field synergy principle for enhancing convective heat transfer—its extension and numerical verifications, Int. J. Heat Mass Tran. 45 (2002) 3849–3856.
[6]Y. Jin, G.H. Tang, Y.L. He, Wen-Quan Tao, Parametric study and field synergy principle analysis of H-type finned tube bank with 10 rows, Int. J. Heat Mass Tran. 60 (2013) 241–251.
[7] O.A. Mohammed Hamid, Bo Zhang, Luopeng Yang, Application of field synergy principle for optimization fluid flow and convective heat transfer in a tube bundle of a pre-heater, Energy. 76 (2014) 241-253.
[8]T.T. Li, Y.L. He, Y.G. Lei, W.Q. Tao, Numerical study of fluid flow and heat transfer in a flat-plate channel with longitudinal vortex generators by applying field synergy principle analysis, Int. Commu. in Heat and Mass. 36 (2009) 111–120.
[9]ANSYS Inc. Fluent 14.5 User's Guide.